Welcome

AltiumLive 2018
University Day

Instructor
John Watson CID
Senior PCB Engineer- Legrand Inc
Design Rules and Advanced Constraints

To instruct designers to be able to create more advanced rules for the PCB, including verification of the rules and the various means for generating the rule constraints.
Agenda

• Introduction
• PCB Design Rules Overview
• Level 1- Component Rule Checks
• Level 2- Schematic ERC Checks & Connection Matrix
• Level 3- BOM PCB Design Checks
• Level 4- PCB Design Rules (DRC)
• Level 5- Multi-Board Project ERC
• Special PCB Design Rules Tools
• Putting it all together
• Conclusion
There stands a Lighthouse
PCB Design Rules Definition and Objective

- Defined

As a set of parameters to verify the accuracy of a PCB design to the desired design criteria. Rules are separate from the objects.
PCB Design Rules Definition and Objective

- Defined
  As a set of parameters to verify the accuracy of a PCB design to the desired design criteria. Rules are separate from the objects

- Objective
  - **NOT** to have a clean ERC/ DRC report.
  - To match the PCB design to the strictest rule set to sustain Integrity in the design and the process
Fundamentals of the PCB Rules System

- Rules are separate from the objects
- Two types of Design Rules Unary and Binary
- Each rule has a priority
- Rules are targeted (scoped) by writing a query
Mistakes with PCB Design Rules

1. Rules Not Setup at all or incorrectly.
2. Rule Scope/constraints are changed to match the design.
3. Rules are not adapted to the type of PCB design.
4. Not following Regulatory Compliance guidelines
5. Not following DFM requirements.
6. Rules are only checked at the “end” rather than throughout the design process.
7. Thinking that these are NOT Rules... but suggestions
The Five levels of PCB Design Rules

- Level 1: Component Rule Checks
- Level 2: Schematic ERC & Connection Matrix
- Level 3: BOM Checks (ActiveBOM)
- Level 4: PCB Design Rules (DRC)
- Level 5: Multi Board Project ERC
Design Rules and Advanced Constraints

Setup
Criteria: Component open for editing
Command: System Preferences > Data Management > Component Rule Check

STATS
• 3 Categories
  - Violations associated with Components
  - Violations associated with Footprints
  - Violations associated with Symbols
• Total 20-Rules setup
• 4-Levels of Reporting
Design Rules and Advanced Constraints

Running Checks

Criteria: Component open for Editing
Command: Tools> Component Rule Check

Any Issues are displayed in the Messages window

- Component Rule Checks
- Schematic ERC & Connection Matrix
- BOM Checks (ActiveBOM)
- PCB Design Rules (DRC)
- Multi Board Projects
Setup

Criteria: Open Schematic
Command: Projects>Project Options> Error Reporting

**STATS**

- 9 Categories
- Violations associated with Buses
- Violations associated with components
- Violations associated Documents
- Violations associated with Harnesses
- Violations associated Nets
- Violations associated Others
- Violations associated Parameters

- Total 88-Rules setup
- 4-Levels of Reporting
Basics- Connection Matrix

Criteria: In the Schematic
Command: Project>Project Option> Connection Matrix

Each Schematic Pin has an Electrical Type

Purpose:
To establish connectivity rules between component pins and net identifiers, such as Ports and Sheet Entries.
Compiler Preferences

Criteria: Open Schematic

Command: System Setup>Schematic>Compiler
Running the Checks

Command: Project>Compile the Project

- Component Rule Checks
- Schematic ERC & Connection Matrix
- BOM Checks (ActiveBOM)
- PCB Design Rules (DRC)
- Multi Board Projects

Any errors are displayed in the Messages window
Interpreting Messages & Locating Errors

Command: Right Clicking on any Error>Cross Probe

Able to:
• Group
• Cross Probe
• Save/Export/Report
• Cross Probe a specific error

- Component Rule Checks
- Schematic ERC & Connection Matrix
- BOM Checks (ActiveBOM)
- PCB Design Rules (DRC)
- Multi Board Projects
Setup

Criteria: Created ActiveBom Document
Command: Files>New>ActiveBOM Document
Setup & Running

Command: BOM Checks > Checks Options

STATS
2 Categories
Violations associated with Design Item
Violations associated with Part Choices
Total 24-Rules setup
4-Levels of Reporting
Overview
Criteria: In a PCB Design
Command: Design > Design Rules

STATS
10 - Major Categories

62 - Minor Categories

ENDLESS Variations of Rules

- Component Rule Checks
- Schematic ERC & Connection Matrix
- BOM Checks (ActiveBOM)
- PCB Design Rules (DRC)
- Multi-Board Project
Activating select Rules

Criteria: In a target PCB Design
Command: Tools>Design Rule Check

- Component Rule Checks
- Schematic ERC & Connection Matrix
- BOM Checks (ActiveBOM)
- PCB Design Rules (DRC)
- Multi-Board Project

Specific Rules can be enabled or disabled
Big area of setup problems
**PCB DRC Preferences- Online DRC**

**Command:** Setup System Preferences>PCB Editor>General

When Online DRC is selected will check the Design DRC rules while the design process is occurring.

- Component Rule Checks
- Schematic ERC & Connection Matrix
- BOM Checks (ActiveBOM)
- PCB Design Rules (DRC)
- Multi-Board Project
Online/Batch Setup

Criteria: In a target PCB Design

Command: Tools>Design Rule Check

- **DRC Online**
  What rules will be run when using DRC Online checks

- **Batch**
  What rules will be run, when rule check is run “manually”.

Important Batch check should be much more detailed with all rules enabled.

Final methods to check design.
PCB DRC Preferences - Violation Display

Command: Setup System Preferences > PCB Editor > PCB DRC Violation Display Preferences

1. Errors are displayed either with colored Overlay Of various Colors and overlay styles.
2. Errors are displayed with a Violation detail graphic with the violating rule details.
Setting up a New Rule

Criteria: In correct Major and Minor Rule Category

Command: >New Rule

- Component Rule Checks
- Schematic ERC & Connection Matrix
- BOM Checks (ActiveBOM)
- PCB Design Rules (DRC)
- Multi-Board Project

Give the Rule a Unique Name and Comment

Explains what the rule will accomplish
Scope, Constraints & Test

**Scope**
- All
- Net
- Net Class
- Layer
- Net and Layer
- Custom Query

**Constraints**
- 2-Levels of Constraints
  - Simple and Advanced
  - Specifics of the rule.

**Test Queries**

- Component Rule Checks
- Schematic ERC & Connection Matrix
- BOM Checks (ActiveBOM)
- PCB Design Rules (DRC)
- Multi-Board Project

**IMPORTANT** Each Category and sub category will have different Requirements
The Priority Rule
First Run Specific Rules
Before General rules
Running Rules Checks (Batch)
Criteria:
Command: Tools>Design Rule Check>Report Options

Should be used as a final check of the Design Report Options
Double Check the settings.
Usually keep everything selected

Rules to Check
Specific rules under Batch are activated.

To Run DRC Check
<Run Design Rule Check>
Any Errors will be in a detailed Design Rule Verification Report OR it will also be in the Message window like all other errors.
Navigating Error Report

- Component Rule Checks
- Schematic ERC & Connection Matrix
- BOM Checks (ActiveBOM)
- PCB Design Rules (DRC)
- Multi-Board Project

By Cross Probing will show the violation on the PCB with the Overlays and Violation Details
Setup
Criteria: In a Multi-Board Project
Command: Project Options > Error Reporting

**STATS**
- 2 Categories
- Violations associated with Connections
- Total 5-Rules to setup
- 4-Levels of Reporting
Running ERC Checks

Criteria: In a Multi-Board Project

Command: Design>Run ERC

All Errors will show up in the Message Window

With the ability to sort and cross probe any problem
Special PCB Design Rules Tools
Criteria: PCB Design
Command: Design> Run Wizard

Recommend being able to setup rules manually before using this tool

- Rule Wizard
- PCB Rules and Violation Panel
- PCB Filter Panel
- Custom Queries with Helper
- Custom Queries with Builder
Rule Wizard Con
Criteria: PCB Design
Command: Design> Run Wizard

Step 4 - Choose the Rule Priority

Step 5 - The Rule is Complete
Design Rules and Advanced Constraints

Special PCB Design Rules Tools
Criteria: In PCB Design
Command: View>Panels> PCB Rules and Violations

Power tool to browse the enabled design rules and violations in the current board layout workspace.

1. Rule Classes - Design rules grouped by classes.

2. Rules - The individual design rules of the selected class. Can be run by right clicking on them.
Special PCB Design Rules Tools

Criteria: In PCB Design
Command: View>Panels> PCB Rules and Violations

3. Violations - Details of each rule violation reported by DRC Ability to jump to and magnify the violation. Then to Waive it

4. Waived Violations - Details of each waived rule violation Who Waived the violation and why. No more simply turning rules off. Each error must be fixed or waived. 

*Very Important*

- Rule Wizard
- PCB Rules and Violation Panel
- PCB Filter Panel
- Custom Queries with Helper
- Custom Queries with Builder
Special PCB Design Rules Tools

Criteria: In PCB Design
Command: View> Panels> PCB Filters

Provides controls to construct filters through the creation of logical queries. Helps to define scoping for PCB design rules.

3-Parts of the Filter
1. Object
2. Layer
3. Filter

Becomes a powerful DRC tool in #4. Can create a rule directly into the DRC setup, Helper or Builder.
Special PCB Design Rules Tools

Criteria: PCB Design
Command: View>Panels> PCB Filters>Builder

Tool allows you to create a query for targeting specific objects in the design document by simple construction of a string of ANDed and/or ORed conditions.

Can build the Query level by level based on certain conditions and condition Value.

Very Powerful Tool in Altium options are absolutely endless.

- Rule Wizard
- PCB Rules and Violation Panel
- PCB Filter Panel
- Custom Queries with Helper
- Custom Queries with Builder
Special PCB Design Rules Tools

Criteria: PCB Design
Command: View>Panels> PCB Filters>Builder

- Rule Wizard
- PCB Rules and Violation Panel
- PCB Filter Panel
- Custom Queries with Helper
- Custom Queries with Builder

Increase capabilities and function from the Query Helper.
- PCB Functions
- PCB Objects List
- Systems Functions

Fortunately as you build the query Altium helps you along.

Would help to build a strong understanding with Query Language and how to use it
Putting it all together
Command: Place>Directives>Generic No ERC
Putting it all together

Command: **Place>Directives>Differential Pairs**
Putting it all together

Command: **Place>Directives>Parameter Set**

Parameter Set with Blanket

Clearance Design Rule
Putting it all together

Command: Place>Directives>Compile Masks

Compile Mask

Purpose:
For various reason, if a certain portion of the circuit does not want to be checked when ERC are run.

A Compile mask is used to ignore that circuit.

So as the circuit is being developed and you know it is good than it can be compile masked over.
Thank You

AltiumLive 2018

Questions?